

PRESSURE DROP ANALYSIS OF 1.6L CAR AIR INTAKE SYSTEM

MOHAMAD SAFWAN BIN AHMAD LOTHFY

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

Today, air intake system and filter play major role in getting good quality air into automobile engine. The intake system has improves the combustion efficiency and also reduces air pollution. This paper focuses on the pressure drop analysis of 1.6L car air intake system. The thesis describes the CFD analysis techniques to predict the pressure drop and identify the critical locations of the components. The air intake system is work as to supply the engine with clean air and correct amount for the required air to burn in the manifold chamber. This research is to analyze the model and pressure drop of Proton Waja intake system. The thesis describes the CFD analysis techniques to predict the pressure drop and identify the critical locations of the components. 3D viscous CFD analysis will carry out for an existing model to understand the flow behavior through the intake system, air filter geometry and filter media. Results obtain from CFD analysis of the existing model show good correlation with experimental data. Based on existing model CFD results, it can show the airflow in the intake system and the pressure drop will see by pressure visualization. The time and cost are reducing by using 3D CFD analysis for air intake system in automobile industry.

ABSTRAK

Dewasa ini, sistem pengambilan udara dan penapisan memainkan peranan penting dalam mendapatkan kualiti udara yang memasuki injin automobil. Sistem ini meningkat kecekapan pembakaran dan mengurangkan pencemaran udara. Projek ini memberi fokus kepada analisis tekanan yang jatuh dalam sistem pengambilan udara bagi kereta 1.6 liter. System ini juga berfungsi membekalkan udara bersih dan jumlah udara yang diperlukan untuk pembakaran di dalam kebuk pancarongga. Penyelidikan ini bertujuan mengkaji model dan tekanan yang jatuh untuk sistem pengambilan udara bagi kereta Proton Waja. Analisis CFD dianalisis ke atas model bagi mamahami corak sistem pengaliran udara melalui sysem pengambilan udara luar, geometri penapisan udara dan media penapisan. Keputusan yang diperolehi dari analisa model CFD menunjukkan bahawa corak sistem ini mempunyai perkaitan yang baik. Daripada keputusan yang dikeluarkan oleh CFD, ia menunjukkan pengaliran udara dalam sistem ini dan melihat bagaimana tekanan jatuh melalui visual takanan. Dengan menggunakan analisis dari CFD, ia dapat mengurangkan kos dan masa untuk mengkaji sisten pengambilan udara di dalam industri automotif.

TABLE OF CONTENTS

	Page
SUPERVISOR’S DECLARATION	ii
STUDENT’S DECLARATION	iii
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xi
LIST OF FIGURES	xii
LIST OF SYMBOLS	xiii
LIST OF ABBREVIATIONS	xiv
 CHAPTER 1 INTRODUCTION	
 1.1 Background of Study	1
1.3 Problem Statement	2
1.3 Objectives of Study	3
1.4 Scopes of Study	3
 CHAPTER 2 LITERATURE REVIEW	
 2.1 Introduction	4
2.2 Air Intake System	4
2.3 Bernoulli Equation	6
2.3.1 Major Loss	7
2.3.2 Minor Loss	7
2.3.3 Total Pressure	8
2.4 Mass And Volume Flow Rate	8
2.5 Computational Fluid Dynamics (CFD)	9
2.5.1 Simulation Benefits	10

CHAPTER 3 RESEARCH METHODOLOGY

3.0	Introduction	11
3.1	Flow Chart	12
3.2	Data Collecting	13
3.3	Structural Modeling	14
3.4	Simulation	19
	3.4.1 Boundary Condition	19
	3.4.2 Mesh	20

CHAPTER 4 RESULTS AND DISCUSSION

4.1	Introduction	21
4.2	Velocity Visualization	22
4.3	Pressure Visualization	23
4.4	Simulation Analysis	25
	4.4.1 Sample Of Calculation	26
4.5	Complete Result Of Simulation	27
4.6	Pressure Drop Result	33
4.7	Pressure Drop Analysis	34

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1	Conclusions	36
5.2	Recommendations	37

REFERENCES

38

APPENDICES

A Project Gant Chart

39

B Technical Report

42

List of Tables

Table No.	Title	Page
4.1	Simulation result at entering pipe for 1000 rpm	25
4.2	Simulation result at exit pipe for 1000 rpm	26
4.3	Simulation result at entering pipe for 2000 rpm	27
4.4	Simulation result at exit pipe for 2000 rpm	27
4.5	Simulation result at entering pipe for 3000 rpm	28
4.6	Simulation result at exit pipe for 3000 rpm	28
4.7	Simulation result at entering pipe for 4000 rpm	29
4.8	Simulation result at exit pipe for 4000 rpm	29
4.9	Simulation result at entering pipe for 5000 rpm	30
4.10	Simulation result at exit pipe for 5000 rpm	30
4.11	Simulation result at entering pipe for 6000	31
4.12	Simulation result at exit pipe for 6000 rpm	31
4.13	Simulation result at entering pipe for 7000 rpm	32
4.14	Simulation result at exit pipe for 7000 rpm	32
4.15	Result of pressure drop based on engine speed	33

LIST OF FIGURES

Figure No.	Title	Page
2.1	An illustration of the air intake system structure	5
3.1	Flow chart	12
3.2	Proton Waja air intake system	13
3.3	3D of air intake system	14
3.4	intake pipe	14
3.5	Air pipe	15
3.6	Upper box	15
3.7	Down box	16
3.8	Filter	16
3.9	After assemble	17
3.10	Orthographic view	18
3.11	Boundary condition	19
3.12	Mesh	20
4.1	Velocity visualization	22
4.2	Pressure visualization	23
4.3	Visual of pressure at filter box	24
4.4	Full pressure visualization of air flow	24
4.5	Pressure drop graph based on engine speed	33
5.1	Flow bench machine	36

LIST OF SYMBOLS

μ	Dynamic viscosity
V	Fluid velocity
ρ	Density
ν	Kinematic viscosity
m	Molar mass
\dot{m}	Mass flow rate
A_c	Area of the nozzle
A_s	Area of the wall
f	Friction factor
K_L	Loss coefficient
h_L	Head loss
Re	Reynolds number
g	Gravity
D	Pipe diameter
Δp	Pressure loss

List of Abbreviations

FYP	Final year project
vs	Versus
3-D	Three Dimension

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This thesis focuses on the intake system pressure drop analysis using CFD analysis results and experiment. In the end of this project, the pressure drop of air flow cross the air intake can be analyzed. The engine of a car needs air for the combustion process in the cylinders. Air intake system and filter play major role in getting good quality air into automobile engine. It improves the combustion efficiency and also reduces air pollution. For this thesis, the Proton Waja 1.6 Air intake system has been choosing to analyze. The main function of an air intake system is to supply the engine with clean air and correct amount for the required air to burn in the manifold chamber. Air enters the filter through dirty pipe and inlet side plenum, which guides the flow uniformly through the filter media. Optimum utilization of filter can significantly reduce the cost of filter replacements frequently and keep the filter in use for longer time. To optimize intake system and filter, thorough understanding of flows and pressure drop through the system is essential. Computational Fluid Dynamics (CFD) is considered to be the most cost effective solution for flow analysis of intake system along with filter media. Air intake systems employ specially-shaped intake tubes designed to straighten airflow as much as possible while looking great in engine compartment. These pipes are typically mandrel-bent, a process that doesn't crimp the pipe diameter at the bend.

1.2 PROJECT PROBLEM STATEMENT

Car air intake systems allow the car to breathe easier creating more horsepower and greater gas mileage. Grabbing a high-performance air intake is the quick and easy route to several benefits, including:

- An instant increase in horsepower
 - A noticeable boost to your throttle response
 - Improved fuel economy
 - A long-life, washable performance air filter
 - Specialized engineering that's fine-tuned to your specific vehicle
 - Straightforward, simple installation virtually anyone can complete
- Horsepower increase from a performance air intake

The flow efficiency of the intake system has a direct impact on the power the engine is able to deliver. This project is to analyze the pressure drop of Proton Waja intake system. If the flow in the air intake determined to have less turbulent flow and decrease the wake projection and there is less pressure drop across the intake system it will increase the efficiency of combustion of the air in the intake system. The CFD will be use to analyze the internal flow of air intake and get the initial result. From the analysis, the value of pressure drop in certain rpm of engine power can be determined. The difference speed of air flow based on the lower until maximum rpm of engine will be used.

For performance intake draws in a higher volume of air which may be much cooler, your engine can breathe easier than with a limiting stock system. With combustion chamber filled by cooler, oxygen-rich air, fuel burns at a more efficient mixture. It will get more power out of every drop of fuel when it's combined with the right amount of air. With more air in the chamber, it can also burn more fuel than before. That's how a performance intake puts power at the pedal for reducing air temperatures, balancing fuel mixtures and providing more air for combustion.

1.3 OBJECTIVES

The objectives of this project are:

- i. To determine and analyze the pressure drop in the air intake system.
- ii. To analyze the model of Waja 1.6 Air Intake System using the CFD
- iii. To estimate flow rates of the air intake system across the intake system.
- iv. To analyze the air flow affected by the minor losses.

1.4 SCOPES OF STUDY

The scope of this project will comprise the boundaries of project study. The pressure drop analyses of air intake system are wide range of study. Many characteristic should be bound in order to make this project achieve the objectives. First of all, the study of this project is using Bernoulli Equation to determine the pressure drop in calculation. This equation is very useful to identify the velocity, density and pressure of the air flow. Furthermore, this project study using the Solidworks to design the air intake system model of the Proton Waja 1.6L. The CFD will be use to analyze the internal flow of air intake and get the result in difference speed of engine from minimum until maximum. The speed of engine start from 1000 rpm until 7000 rpm will be use in the simulation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter presents a review of literature on the efforts relating to the pressure drop analysis of air intake system. It attempts to establish what the factors are affecting the performance of intake system and how this intake system affected the car engine performance. The discussions are focus on the flow management in the intake system as a tool to improve the car engine performance.

2.2 AIR INTAKE SYSTEM

For an engine equipped with a carburetor, this is pretty straightforward. Air comes in the air filter housing, passes through the air filter, into the carburetor where the fuel is mixed with it. Then it passes through the intake manifold and is drawn into the cylinders [1]. The most advanced part of the system was an Air Temperature Sensor in the air intake. It was used to measure the air temperature and, by opening and closing a flap, allow cool air in through the air horn or heated air piped in from around an exhaust manifold [2]. This was to prevent carburetor icing that would cause the car to stall and die out. It also facilitated vaporization of the fuel into the air stream [1].

In a fuel-injected car it's a whole different ball game. Air is drawn in through the air intake. This is usually a long plastic tube going into the air filter housing. The reason the intake tube is long is to get the air moving in a fairly steady, coherent stream. It then passes through the air filter and then through an Air Flow Meter [3].

The intake system of an engine has three main functions. Its first and usually most identifiable function is to provide a method of filtering the air to ensure that the engine receives clean air free of debris. Two other characteristics that are of importance to the engineers designing the intake system are its flow and acoustic performance [5]. The flow efficiency of the intake system has a direct impact on the power the engine is able to deliver. The acoustic performance is important because government regulations dictate the maximum noise level that vehicles can make during a pass-by test. The speed of air generated by the intake system can be a significant contributor to this pass-by noise and separated flow [6]. It may be noted that since the loss pressure from the intake duct towards atmosphere, this paper assumes the inlet is at the intake manifold and air filter duct and the outlet is at atmosphere.

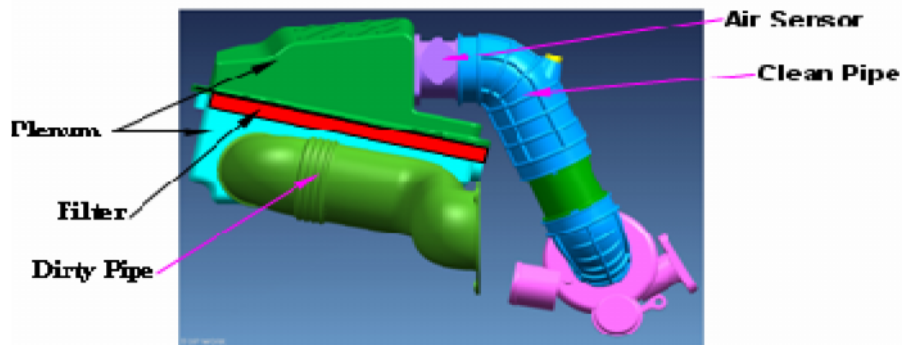


Figure 2.1: An illustration of the air intake system structure

Source: Ravinder Yerram and Nagendra Prasad
Quality Engineering and Software Technologies (QUEST), Bangalore

Air intake systems employ specially-shaped intake tubes designed to straighten airflow as much as possible while looking great in your engine compartment. These pipes are typically mandrel-bent, a process that doesn't crimp the pipe diameter at the bend. Special care is given to locating the intake tube, air box and filter in the position that best fosters maximum performance. The materials used are also selected with optimum engine conditions in mind.

The fundamentals of installing a performance air intake on your vehicle not only is a performance air intake one of the most essential upgrades to your vehicle, it happens to be one of the easiest additions to install. With little more than a common socket set, a couple of screwdrivers and half an hour's time, you can have your new air intake in place and ready to roar [5].

Detailed instructions are included with every intake kit. These instructions go through the simple process of removing your stock intake system (including the tube and air box), and installing the new air intake in just a few minutes to stock mounting positions. No cutting, drilling or other modifications are required [2].

2.3 BERNOULLI EQUATION

The Bernoulli equation is a useful equation as it relates pressure changes to velocity and elevation changes along a streamline. Streamlines are lines drawn in a flow field so that at a given instant they are tangent to the direction of the flow at every point in the flow field. Since they are tangent to the velocity vector at every point in the flow field, there can be no flow of fluid across a streamline [4].

$$\frac{P_1}{\gamma} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\gamma} + \frac{V_2^2}{2g} + Z_2 + h_L \quad (2.1)$$

The Bernoulli equations give correct results when certain restrictions are applied. These are as follows:

1. Steady flow
2. Incompressible flow
3. Frictionless flow
4. Flow along a streamline.

$$\gamma = \rho g$$

2.3.1 Major Loss

The most pipe or duct system consists of the straight pipe at this point head loss due to viscous effect *Major losses*, $h_{L\ major}$ can be determined by equation:

$$Major\ losses, h_{L\ major} = f \frac{\ell}{D} \frac{V^2}{2g} \quad (2.2)$$

Friction factor, f to be determine using Moody chart. Using the Reynolds number and for the plastic surface of the AIS we look the graph curve at the graph smooth line [4]. Because the roughness, ε . For the laminar developed flow, the value of f is simply:

$$f = \frac{64}{Re}$$

2.3.2 Minor Loss

In the intake system we found the system of the pipe more than a straight pipe. These additional components (valves, bends, tees, and the like) add the overall head loss of e system [8]. Such losses are generally termed *Minor losses*, with the corresponding head loss denoted $h_{L\ minor}$.

$$Minor\ losses, h_{L\ minor} = K_L \frac{V^2}{2g} \quad (2.3)$$

The most common method used to determine these head losses or pressure drops is to specify the loss coefficient, K_L which is defined as:

$$K_L = h_{L\ minor} \cdot \frac{2g}{V^2} = \frac{\Delta p}{\frac{1}{2} \rho V^2} \quad (2.4)$$

Pressure drops, Δp :

$$\Delta p = K_L \frac{1}{2} \rho V^2 \quad (2.5)$$

2.3.3 Total Pressure

Total Pressure is obtained when the flowing fluid is decelerated to zero speed by a frictionless process [4]. In an incompressible flow the Bernoulli equation can be used to relate the changes in speed and pressure along a streamline for such a flow. Neglecting elevation, then equation becomes:

$$\frac{P_1}{\rho} + \frac{V_1^2}{2} = \frac{P_2}{\rho} + \frac{V_2^2}{2} \quad (2.6)$$

$$\frac{P_1}{\gamma} + \frac{V_1^2}{2g} = \text{constant}$$

If the static pressure P_1 is at a point in the flow where the speed is V_2 , then the total pressure P_t , where the stagnation speed, V_t , is zero, then the equation becomes:

$$P_t = P_2 + \rho \frac{V_2^2}{2} \quad (2.7)$$

2.4 MASS AND VOLUME FLOW RATE

The amount of mass flowing through a cross section per unit is called the mass flow rate and its denoted by \dot{m} [8]. The dot over a symbol is used to indicate *time rate change*.

$$\dot{m} = \rho V_{avg} A_c \quad (2.8)$$

We defined the average velocity V_{avg} average value across the entire cross section of the pipe, where A_c is the area of the cross section normal to the flow direction.

The volume of the fluid flowing through a cross section per unit time is called volume flow rate, \dot{V} or Q .

$$\dot{V} = V_{avg}A_c = VA_c$$

The mass and volume flow rates are related by:

$$\dot{m} = \rho\dot{V}$$

2.5 COMPUTATIONAL FLUID DYNAMICS (CFD)

Air was used as fluid media, which was assumed to be steady and incompressible. High Reynolds number k- ϵ turbulence model was used in the CFD model. This turbulence model is widely used in industrial applications. The equations of mass and momentum were solved using SIMPLE algorithm to get velocity and pressure in the fluid domain. The assumption of an isotropic turbulence field used in this turbulence model was valid for the current application. The near-wall cell thickness was calculated to satisfy the logarithmic law of the wall boundary. Other fluid properties were taken as constants [11]. One of the most important requirements before a CFD computation can be performed is the availability of a suitable grid. Inability to construct a grid quickly and reliably often rules out a CFD analysis. Linear methods such as the Panel Method need only a grid on the body surface (and road). Generation of the grid on the surface of a real vehicle so as to correctly capture the critical flow phenomenon is not a trivial problem. The Computer-Aided Design (CAD) surface definition data created for body panel manufacture in the industry are helpful in generating such grids [12]. The nonlinear CFD methods (Euler, NS) need a body-surrounding spatial grid to solve the partial differential equations. The inner boundary of this grid is the body surface and the outer boundary is the bounding surface of a sufficiently large computational domain around the body

The aim of CFD is to resolve the equations that drive theoretically every kind of flow:

- The continuity equation
- The momentum equations
- The energy equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0 \quad (2.9)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \quad (2.10)$$

where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass, E is the total energy per unit mass, Q_H is a heat source per unit volume,

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial}{\partial x_k}((\rho E + P)u_k + q_k - \tau_{ik}u_i) = S_k u_k + Q_H \quad (2.11)$$

is the viscous shear stress tensor and q_i is the diffusive heat flux.

2.5.1 Simulation Benefits

Technical Advantages

- Faster evaluation of new ideas, products and processes
- New insights into your process and performance
- Maximise effectiveness of your manufacturing resources
- Save time and cost, and get better results

Business Advantages

- Reduce risk and increase confidence in technical projects
- Increase customer confidence
- Increase credibility with customers
- Win more business

CHAPTER 3

METHODOLOGY

3.0 INTRODUCTION

In this project, simulation will be conducted by varying the velocity of air based on the speed of engine. Research and approach will be described clearly in flow chart, procedures, dimension measurements, modeling and simulation. The collected data from the simulation will be used for further analysis.

3.1 FLOW CHART

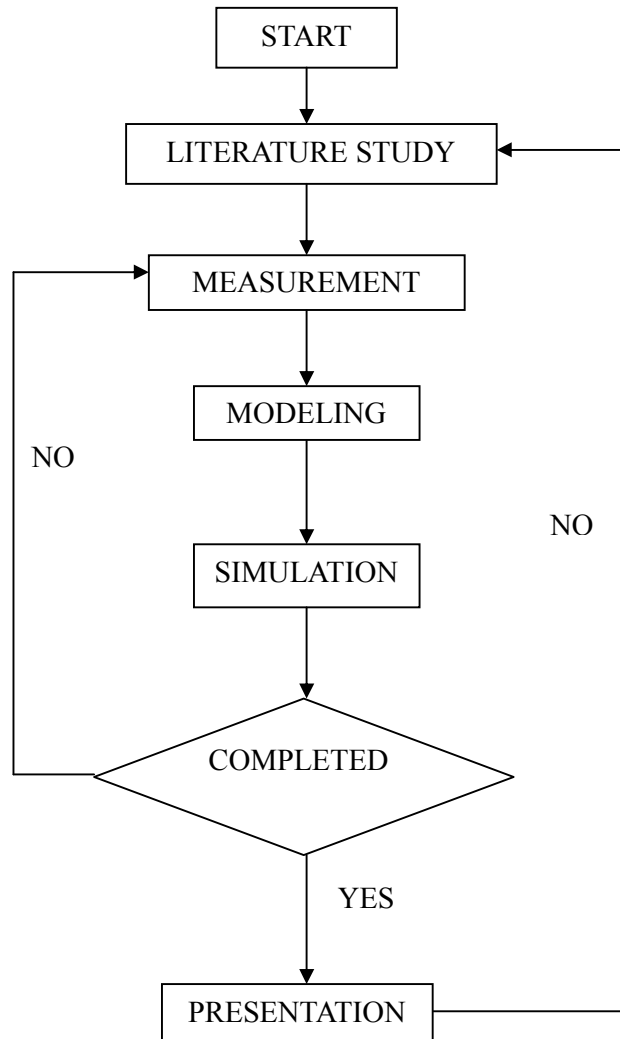


Figure 3.1: (Flow Chart)

3.2 DATA COLLECTING



Figure 3.2: Proton Waja air intake system

The data of dimension for air intake system of Proton Waja was collecting from measuring then modeling the body by SolidWorks software. Data collecting of dimension as accurate as possible is very important for air intake to simulating the model in CFD.

To measure the area, venire caliper had been used.

Inlet cross section area = 0.03756m^2

Thickness of plastic = 29.5 mm

Porosity (filter) = 0.85

3.3 STRUCTURAL MODELING – USING SOLIDWORK

After measure all dimension of the air intake, the model has been design by using solidwork software. Every single part of air intake has been drawn and finally all part will assemble.

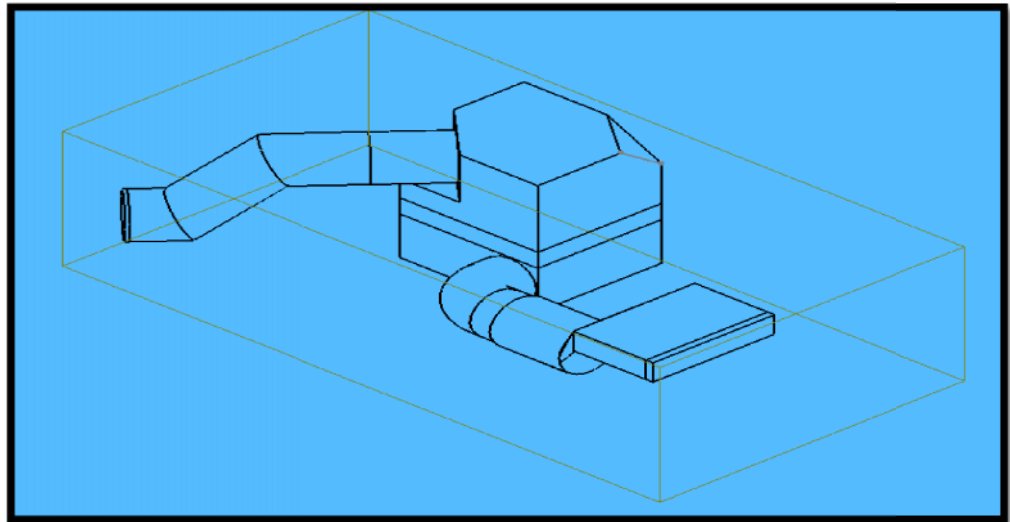


Figure 3.3: 3D of air intake system

The first part: intake pipe

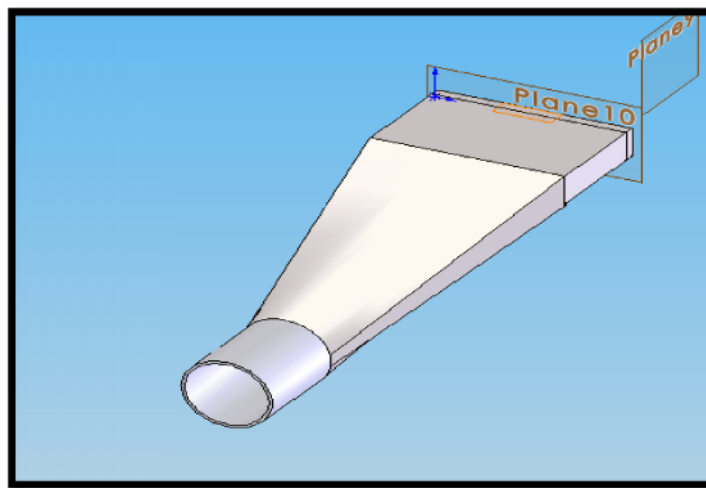


Figure 3.4

Air Pipe

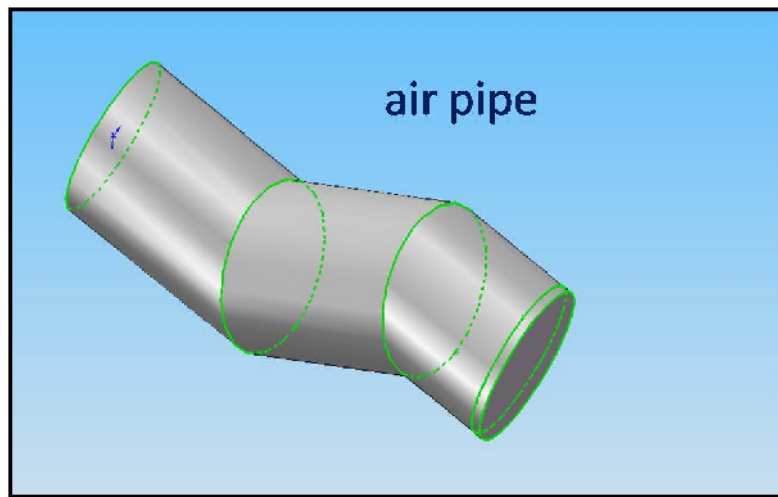


Figure 3.5

Upper box

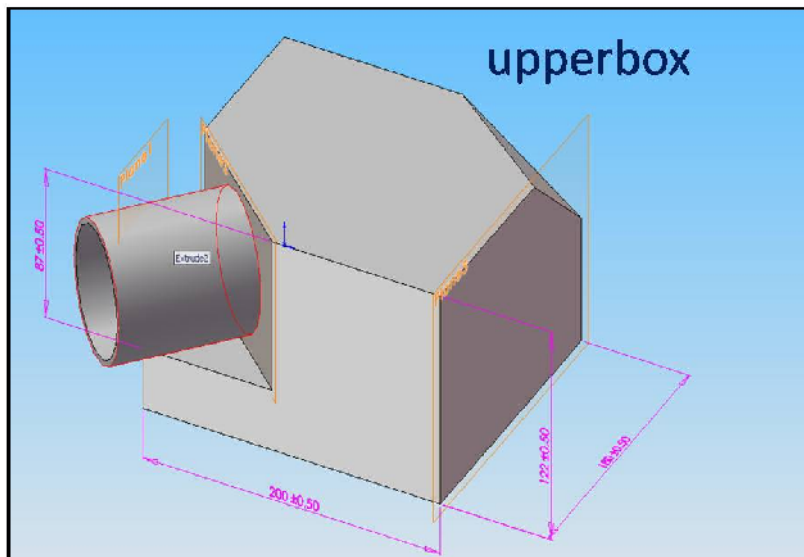


Figure 3.6